Brake Squeal Background

Brake noise has been a challenging issue for the automobile industry for decades. Generally accepted categories include:

- **Low-frequency noise (100 Hz to 1,000 Hz):** groan and moan noise
- **Low-frequency squeal (1,000 Hz to 7,000 Hz):** coupling out-of-plane modes of rotor with bending modes of pads
- **High-frequency squeal (8,000 Hz to 16,000 Hz):** coupling in-plane modes of rotor with bending modes of pads; brake disk develops friction-induced oscillations, which are heard as “brake squeal”

Although all brake squeal mechanisms are not fully understood, it is commonly accepted that brake squeal is initiated by instability due to friction forces, leading to self-excited vibrations. Vibration modes of the disk and pad couple to each other as the coefficient of friction between the pad and disk increases. To predict the onset of instability, engineers can perform modal analysis of the prestressed structure. An unsymmetric stiffness matrix is a result of the friction coupling between the brake pad and disc; this may lead to complex eigenfrequencies. If the real part of the complex frequency is positive, then the system is unstable as the vibrations grow exponentially over time.

---

White Paper

Breakthrough in Brake Squeal Prediction Helps to Eliminate Noise Problems Early in Design Process

By Mike Hebbes
Regional Technical Manager
ANSYS, Inc.

Engineers have succeeded in recent years in reducing many types of automobile noise. However, friction-induced squeal in automotive brakes is an increasing source of customer complaints. Currently, brake noise is one of the top warranty issues that automobile original equipment manufacturers (OEMs) and suppliers face. A recent breakthrough is an integrated approach to brake squeal prediction: It incorporates bidirectional computer-aided design (CAD) connectivity, automated meshing and connectivity, flexibility to use a linear and/or nonlinear solver, parametric and sensitivity studies, and a wide range of graphical outputs. This integrated approach substantially reduces setup time, correlates well with physical testing, maintains in-sync models with production and the supply chain, and makes it possible to automatically evaluate a large number of design alternatives to quickly identify the optimal design. The new method enables first-time virtual design of cost effective, quiet braking systems for multiple platforms in far less time than is possible with conventional methods.

Brake Squeal Background

Brake noise has been a challenging issue for the automobile industry for decades. Generally accepted categories include:

- Low-frequency noise (100 Hz to 1,000 Hz): groan and moan noise
- Low-frequency squeal (1,000 Hz to 7,000 Hz): coupling out-of-plane modes of rotor with bending modes of pads
- High-frequency squeal (8,000 Hz to 16,000 Hz): coupling in-plane modes of rotor with bending modes of pads; brake disk develops friction-induced oscillations, which are heard as “brake squeal”

---

Automobile OEMs and suppliers want to reduce warranty costs by delivering quieter brakes. To accomplish this goal, they need the ability to evaluate proposed brake designs to determine propensity to squeal, all prior to building expensive and long-leadtime prototypes.
Traditionally, suppliers had to rely excessively on experimental testing to identify and fix squeal problems. This not only greatly increases brake cost, it leaves many potential operating conditions untested, since experiments can be conducted only with a small set of combinations of operating and design parameters. As a result, only a small portion of the design space is studied while developing brakes. Suppliers continue to face huge warranty issues as brakes squeal in the field when subjected to combinations of operating conditions, pad wear, manufacturing tolerances and design parameters for which they were not tested.

The traditional approach requires manual creation of connections between the rotor and friction material. It necessitates matching nodes at the sliding interface, which, in turn, requires a long, tedious, manual process for mesh generation and makes it impractical to perform parametric studies, in which a large number of different designs are evaluated to determine the one that produces the least noise.

**Design Geometry**

In the new integrated approach, the engineer imports the CAD model into ANSYS® Workbench™ — a single, unified environment in which the entire process of geometry manipulation, meshing, simulation, post-processing and optimization is performed. Design parameters from the CAD model are maintained intact throughout this process, so they can be used for parametric analysis as part of optimizing the design. To reduce solution times, all components can be simplified by removing details that have little or no impact on analysis.

The CAD model of the brake assembly used for production simulations is fully parameterized: It incorporates component-level models such as the pad assembly, caliper, rotor and knuckle, which have manufacturing variability. The individual component dynamic response must be calibrated with simulation tools, and then component-level geometry and material property results are included in the system model. The rotor can be modeled by taking advantage of cyclic symmetry to reduce computational workload.

ANSYS SpaceClaim Direct Modeler — one of the tools in the integrated environment — provides a unique, intuitive and powerful way to manipulate the imported CAD models. The Direct Modeler removes the CAD associativity but allows for creating geometric parameters with neutral formats. The geometric model becomes completely dynamic, allowing the user to move, stretch, add and remove features. All changes to the CAD model occur in real time on the screen, providing instant feedback on a design. Direct Modeler is integrated with the ANSYS Workbench platform to allow rapid design changes and parametric optimization.
Material, Contacts and Joints

The next step is adding material properties, contacts and joints to the model. ANSYS Workbench makes it possible to create a locked-down library of materials properties that can be shared among analysts. After the initial import, ANSYS mechanical tools automatically detect and perform setup for contacts or joints between parts of an assembly. The contact settings and options can be modified, and additional manual contact definitions can be added. Each contact or joint is easily identified using the graphical tools provided in the environment. Users can drill down to a detailed view of each contact and joint to capture the physics.

Meshing

ANSYS meshing technology provides multiple methods to generate a pure hex or hybrid hex/tet mesh, depending on the analysts’ requirements. The brake assembly meshes shown in Figure 5 use a combination of hex meshing methods including sweep, thin sweep, MultiZone and hex-dominant. Automated sweep meshing automatically detects and meshes sweepable bodies with a hex mesh. Thin solid sweep meshing quickly generates a hex mesh for thin solid parts that have multiple faces as source and target. MultiZone sweep meshing uses automated topology decomposition behind the scenes to automatically create a pure hex or mostly hex mesh on complex geometries. Hex-dominant meshing uses hex elements on the boundary of a chunky part with a hybrid hex, prism, pyramid, tet mesh internally.

Analysis Options

A major challenge of brake squeal prediction is capturing the structure’s linear behavior based on its prior linear or nonlinear preloaded status. Linear perturbation analysis is used to solve a linear problem from this preloaded stage. The linear perturbation analysis is essentially automated in ANSYS Mechanical™ software. Typically, in the nonlinear analysis, the Newton–Raphson procedure is applied. The tangent matrix from the Newton–Raphson analysis can be used in the linear perturbation analysis to obtain the preloaded solution, since the linear stiffness matrix without preloading may not provide a solution that is as accurate as a non-prestressed method.

A fully nonlinear prestressed modal analysis is the most accurate method to model brake squeal. It uses a nonlinear static solution to establish the initial contact and to compute the sliding contact between the pads and disc. The applied loads and rotation of the disc create the prestressed effect, and friction contact generates an asymmetric stiffness matrix during static structural analysis. A forced frictional sliding contact analysis is used as an additional load step to model steady-state frictional sliding between a brake pad and
a rotating disc with different velocities. The ANSYS Mechanical CMROTATE command is an effective way to define the velocities on the contact and target nodes of the element component, which are used to determine the sliding direction for the rest of the analysis. The CMROTATE command imparts a tangential sliding velocity at the contact element nodes on the brake disk, causing a sliding contact condition between the brake pad and disk. Then linear perturbation analysis generates the element stiffness matrices, which are generally unsymmetric. The second phase of the linear perturbation analysis is a QR damped or unsymmetric modal analysis. The eigensolver uses the unsymmetric stiffness matrix generated in the contact elements and may produce complex eigenfrequencies.

A partial nonlinear perturbed modal analysis is used when a nonlinear solution is required to establish contact but linear analysis can be used to compute the sliding contact. A static contact analysis is performed to establish the contact conditions. A prestressed matrix is generated under external loading. Next, the first phase of the linear perturbation analysis is performed using the CMROTATE command to establish forced sliding contact. Then the second phase of the linear perturbation analysis, which is a QR damped or unsymmetric modal analysis, is performed. The eigensolver uses the unsymmetric stiffness matrix generated in the contact elements and may produce complex eigenfrequencies.

A linear non-prestressed modal analysis is effective when large deflection or stress-stiffening effects are not critical. This method requires less run time than the previous two methods because a nonlinear base solution is not required and is ideal for performing many parametric simulations. This method involves a partial solution followed by a linear QR damped or unsymmetric eigen solution. The first step is performing a partial solution to generate the element stiffness matrices. The contacts stiffness matrix is based only on the initial contact status and predefined sliding direction through the CMROTATE command. Then a QR damped or unsymmetric modal analysis is performed using the partial solution procedures. The final step is expanding the modes and post-processing the results.

Results

Post-processing and report generation are automated in ANSYS Mechanical. The results of a perturbation analysis show the damped frequencies for each mode number and the stability or real part of eigenvalues. The positive real part of the coupled mode induces instability in the system and may be a source of brake noise or squeal. Figure 6 shows that mode 9 has a positive real component and is a potential source of brake squeal. The analysis results also include the mode shapes that can provide useful diagnostic information to help change the design to eliminate instability. Additional results that can be generated include complex eigenvector animations and RST modal assurance criteria (RST-MAC).
Adding More Physics

The physics that can affect brake noise go beyond structural statics and dynamics. For example, fluid flow around the brake is influenced by both vehicle speed and geometry of the surrounding component. This fluid flow can affect the temperatures of the brake components, which, in turn, can affect thermal stresses that impact structural analysis. Computational fluid dynamics (CFD) simulation can be used to analyze fluid flow around the brakes and effects on component temperature. In turn, component temperatures can be used in a thermal analysis that determines resulting thermal stresses. The operation of vehicle control and hydraulic systems also play an important role in brake operation and can be included in the simulation. New features include squeal damping, solver development and parallelization.

Design Optimization

ANSYS simulation tools operate in the Workbench environment, which makes design optimization a very simple extension of a single simulation. When you change a geometry parameter, ANSYS meshing tools are smart enough to re-apply the previous setup, including mesh distributions specific to the changed entities. ANSYS solvers automatically re-apply their setup as well and solve the new model. The post-processor can then regenerate all of the images, tables, animations, reports and more. This enables engineers to easily compare different designs. Parametric persistence makes it possible to automate the optimization process simply by giving Workbench a table of varying parameters and hitting “update.” File transfer, mapping between physics, boundary conditions and other parameters remain persistent during the update.

ANSYS DesignXplorer™ brings optimization for all physics to the Workbench environment, and it supports analyses in which multiple physics are coupled or analyzed independently. The multiphysics simulations are run automatically without human intervention. The integration platform provides seamless data transfer between applications and a process controller that sequentially simulates all design points and collates the outputs. When the user clicks the “update all design points” button, the first design point (with the first set of parameter values) is sent to the parameter manager in the Workbench integration platform. This drives the changes to the model from CAD system to post-processing. The new design point is simulated and output results are passed to the design point table and stored. The process continues until all design points are solved, defining the design space so that it can later be optimized.

In the real world, values of manufacturing, operational or environmental variables such as dimensions, loads, boundary conditions and material properties that are assumed to be fixed in simulation actually vary due to
manufacturing tolerances and other factors. DesignXplorer software’s six sigma analysis uses information about the uncertainty of input parameters to determine the expected output variation. This helps to determine whether or not the design meets robustness requirements. If not, the user can look at the sensitivity plot and other charts to determine which parameters need to be adjusted or tightened to obtain the required robustness. This information can reveal the tolerances that can be relaxed without compromising design.

Conclusion

Automobile OEMs and suppliers want to reduce warranty costs by delivering quieter brakes. To accomplish this goal, they need the ability to evaluate proposed brake designs to determine propensity to squeal, all prior to building expensive and long-leadtime prototypes. The integrated approach described here offers a breakthrough in that it provides all of the tools needed for brake squeal prediction in a single integrated environment. It saves time by reducing the need for data transfer and automating routine analysis tasks.

The new environment also provides the ability to automatically simulate many design points to quickly iterate to a design that meets noise requirements while delivering the best combination of performance, cost, weight, size and other factors. The ability to move directly to the optimal design helps to improve customer satisfaction while reducing time to market and lowering engineering costs.

ANSYS, Inc. is one of the world’s leading engineering simulation software providers. Its technology has enabled customers to predict with accuracy that their product designs will thrive in the real world. The company offers a common platform of fully integrated multiphysics software tools designed to optimize product development processes for a wide range of industries, including aerospace, automotive, civil engineering, consumer products, chemical process, electronics, environmental, healthcare, marine, power, sports and others. Applied to design concept, final-stage testing, validation and trouble-shooting existing designs, software from ANSYS can significantly speed design and development times, reduce costs, and provide insight and understanding into product and process performance. Visit www.ansys.com for more information.

© 2012 ANSYS, Inc. All Rights Reserved.